Computers and Fluids 152 (2017) 164–181

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

A compressive interface-capturing scheme for computation of compressible multi-fluid flows

Cong-Tu Ha[a], Dong-Hyun Kim[a], Warn-Gyu Park[a,*], Chul-Min Jung[b]

[a] School of Mechanical Engineering, Pusan National University, Busan 609-735, South Korea
[b] Advanced Naval Technology Center, NSRDI, ADD, Changwon, Gyeongnam 645-600, South Korea

ARTICLE INFO

Article history:
Received 20 August 2016
Revised 6 April 2017
Accepted 23 April 2017
Available online 24 April 2017

Keywords:
Unsteady compressible multiphase flow
Shock bubble interaction
Free-surface
Interface-capture
Dual-time stepping technique
High-resolution scheme

ABSTRACT

In this paper, a compressive high-resolution interface-capturing scheme is presented for the computation of compressible multi-fluid flows with high-density ratios and strong shocks. The proposed scheme is coupled with a preconditioned dual-time compressible mixture solver for robust and accurate computations over a wide range of Mach numbers. The scheme is simple and relatively easy to implement. It does not require any calculations for the interface curvature and the normal vector. The numerical approximations were implemented on general, structured grids using an implicit MUSCL upwind approach. Validation tests were conducted for a single reversible vortex, advection of an air-water interface, dam-break flow, and air shock-helium bubble interaction. Finally, a three-dimensional gas-lift flow is presented to demonstrate the capability of the present scheme for handling an interface with large jumps in pressure, temperature, and density.

1. Introduction

Numerical simulations of compressible multi-fluid interface flows have been an area of research over the last decades owing to their common presence in a wide range of engineering and environmental applications. Examples of these flows in the field of engineering include free-surface flows in ship hydrodynamics, underwater explosions, gas-oil flows in pipe production systems, exhaust-air flows behind rockets, and separation, extraction, and mixing flows in chemical reactors. The numerical modeling of complex interface-separating compressible multi-fluids and the coupling of the modeling with a flow model without introducing large errors presents significant challenges. An effective numerical scheme should be able to handle a number of different flow features, such as large density ratios, low-speed flows, and large jumps in pressure, temperature, and velocity across the interface. Many different methods have been presented, which can be grouped into mesh-based Lagrangian, meshless Lagrangian, mixed Eulerian–Lagrangian, and Eulerian types. Each has its own advantages and disadvantages [1–3].

In conventional mesh-based Lagrangian methods [4–6], the grid deforms in each time step and aligns with the interface. Lagrangian mesh-based methods have the advantages of presenting a very sharp interface. However, such methods are less applicable in flows with large interface deformations, which can result in either highly distorted meshes or a totally unphysical grid singularity [1,7]. To address this situation, a number of meshless Lagrangian methods, such as the smoothed particle hydrodynamics (SPH) method [8–10], the moving particle semi-implicit (MPS) method [11,12], and the least-squares collocation method [13], etc., can be used. The common feature of these methods is that they construct the numerical formulation without the use of a conventional mesh. The discrete flow is represented by replacing the mesh with a set of arbitrarily distributed particles that carry information about all the physical variables evaluated at their positions. In this manner, the fluid system evolution is purely governed by interactions among these particles. In the context of interface flows, the particles are explicitly associated with one phase; thus, the interface can easily be identified by tracing the particle distribution [9,13]. Although the meshless methods have a natural ability to treat interface flows (e.g., free surface flows [11,14,15], water-sediment flow [16], bubble rising [9], and Rayleigh–Taylor instability [10]), they still suffer from certain shortcomings, such as the relatively large computational cost, difficulties in the treatment of boundary conditions [17], and numerical instabilities due to particle clustering [18]. For a more detailed review of the meshless methods, interested readers are referred to Shadloo et al. [3], Liu and Liu [8], Nguyen et al. [17], and Idelsohn and Onate [19].

Other alternative approaches to overcome the limitations of conventional mesh-based Lagrangian methods include the use of

* Corresponding author.
E-mail addresses: hacongtu@pusan.ac.kr (C.-T. Ha), wgpark@pusan.ac.kr (W.-G. Park).

http://dx.doi.org/10.1016/j.compfluid.2017.04.025
0045-7930/

remapping techniques. These techniques are usually referred to as mixed Eulerian–Lagrangian types, and include marker-and-cell (MAC) [20–23], arbitrary Lagrangian–Eulerian(ALE) [24–27], and front tracking (FT) methods [28–33]. In the MAC methods, the fluids on either side of an interface are marked with marker particles advected in a Lagrangian manner along the flows. The main advantage of MAC methods is that the interface remains sharp. However, these methods are computationally expensive owing to the requirements for dealing with a large number of particles, particularly in three dimensions. Instead of generating fluids on either side of an interface with the marker particles applied through a MAC method; using an ALE or FT method, only the interface is marked using a set of connected marker particles, which makes these latter methods much more efficient than a MAC method. Interested readers can find a state-of-the-art perspective on the mixed Eulerian–Lagrangian types for MAC methods in McKee et al. [23], and for FT methods in Tryggvason et al. [31]. Although mixed Eulerian–Lagrangian techniques can handle a larger amount of mesh distortion, it is still difficult for them to deal with arbitrarily large interface deformations and topological changes [34].

In contrast to Lagrangian types, Eulerian types use a fixed grid, and therefore no mesh distortion occurs. These methods are typically subdivided into interface-tracking (IT) and interface-capturing (IC) methods. In IT methods, the flow equations are combined with a proper mathematical model to reconstruct the interface and calculate fluxes through the faces of the control volumes. A geometric interface reconstruction can be achieved either in an explicit manner, namely, a volume-of-fluid (VOF) tracking method [35–42] or in an implicit manner, namely, a level set (LS) method [43–45]. In a VOF tracking method, the volume fraction of a certain fluid is advected with the flows by means of an interface advection equation. The volume fraction is assumed to be '0' or '1' in a computational cell without an interface, and between '0' and '1' in cells containing an interface. A detailed discussion on VOF tracking methods is presented in [46,47]. The major advantage of a VOF tracking method is that, usually, the mass is rigorously conserved. Unfortunately, a geometric interface reconstruction (i.e., the calculation of the interface curvature and the normal vector) is considerably complicated for implementation in problems involving complex topological changes, such as the merging or the breakup of an interface. In addition, it should be noted that when an interface becomes too thin to be adequately resolved on a grid, VOF tracking methods form "blobby" filaments to locally enforce the mass conservation [36]. These errors decrease the accuracy of the predicted interface location and can lead to stability restrictions. In LS methods, which surpass VOF tracking methods by implicitly reconstructing a geometric interface, an interface is defined through a so-called sign distance function, which is zero at the interface, greater than zero in a certain fluid, and smaller than zero elsewhere. The distance function is advected with the local fluid velocity by solving the scalar advection equation, and the interface is then tracked by applying a re-initialization procedure [44]. LS methods can reconstruct complex interface topologies rather easily, but do not conserve the material volumes [48–52]. Many attempts to reduce the mass loss (or gain) of the LS methods have led to the development of a variety of LS-based methods, including a refined LS grid method [53], a scale-separation method [54,55], an LS and ghost fluid method [56–59], a hybrid particle LS method [49], a coupled discontinuous Galerkin-LS method [60], a coupled LS-FT method [34,61], and a coupled LS-VOF method [52,62–65]. These LS-based methods have proven capable of handling a complex interface. However, their disadvantage is an increase in algorithmic complexity and computational cost. In addition, the lack of mass conservation in LS-based methods without special handling or refinement still appears to be an unresolved issue [55,64].

Interface-capturing methods, which overcome the main drawbacks of the IT methods, describe the behavior of an interface by using only the flow variables. Interface flows are captured by numerically solving the interface governing equations, without reconstructing the interface. IC methods have the following significant advantages. First, they allow arbitrarily large and complex deformations of the interface. Second, they allow interfaces to be dynamically created, such as in cavitating or boiling flows. Third, IC methods can use a unique set of governing equations and a numerical method, rendering them easy to implement. Fourth, they can be formulated naturally in a conservative form, thereby ensuring that the mass of each fluid is conserved. Owing to these advantages, a substantial amount of effort has been invested in improving the major disadvantage associated with IC methods: a largely diffused interface. Recently developed interface capturing techniques include Riemann shock-capturing [66–75], higher-order shock-capturing [76–84], anti-diffusion interface steepening [85–87], tangent of hyperbola interface-capturing [88–90], phase-field lattice Boltzmann (LB) [91–93], pseudopotential LB [94,95], smooth profile [96], and many other approaches [97–99]. However, all of the capturing techniques presented above, with the exception of the schemes developed by Cassidy et al. [72] and LeMartelot et al. [73], use an explicit scheme for the solution of the flow model. As noted in [73,83,100,101], such explicit schemes are particularly sensitive to stability restrictions for long-time simulations. In addition, when these schemes are used for low Mach numbers, their convergence behavior degrades owing to the large disparity between the acoustic and the convective wave speeds, and the solution accuracy deteriorates as a result of a disproportionate scaling of the artificial dissipation [102–104]. Reasonably good results can only be obtained with either a very fine grid resolution, or a small time step [73,75]. Such fine meshes or small time steps are impractical for multi-dimensional applications. To address these restrictions, an implicit dual-time stepping technique [105–107], coupled with a compressive high-resolution interface-capturing scheme is developed in this work. A dual-time stepping approach introduces a pseudo-time derivative in addition to the physical-time derivative. This pseudo-time derivative is used to rescale the disparity between the acoustic and convective wave speeds and remove any linearization and factorization errors at a given physical-time level. Inner iterations are carried out until a pseudo-steady state is achieved. This will effectively yield a time-accurate solution. After the next sequence of inner iterations, the solution is advanced to the next physical-time level.

The present study is organized as follows. In Section 2, the flow model and thermodynamic closures are presented. An implementation of the numerical methods is then presented in Section 3. In Section 4, results of validation tests and computational examples are presented in two- (2D) and three-dimensions (3D) for a range of interface flow problems. Finally, the work is concluded with a brief summary.

2. Governing equations

2.1. Homogeneous multiphase flow model

The flow model describes multiphase mixtures of three fluids in mechanical and thermodynamic equilibrium. It consists of a compressible Navier–Stocks system of equations [106,108,109] coupled with an interface advection equation [71,74] for the volume fraction of one of the three fluids. These equations are written in generalized coordinates as follows:

$$\frac{\partial \hat{q}}{\partial t} + \frac{\partial (\hat{E} - \hat{E}_v)}{\partial \xi} + \frac{\partial (\hat{F} - \hat{F}_v)}{\partial \eta} + \frac{\partial (\hat{G} - \hat{G}_v)}{\partial \zeta} = \hat{S}, \tag{1}$$